Semi-Annual Status Report

on

NASA Grant No. NSG-1021

Large Scale Disturbances Mechanism of Jet Noise Generation

Period covered by this report:
June 1, 1975 to November 30, 1975

Principal Investigator:
Christopher Tam
Department of Mathematics
Florida State University
Tallahassee, Florida 32306

NASA Technical Officer for this grant:
Dr. Jay C. Hardin
ANRD Division
Mail Stop 465
NASA Langley Research Center
Hampton, Virginia 23665

ASA-CR-146018) LARGE SCALE DISTURBANCES
CHANISM OF JET NOISE GENERATION
miannual Status Report, 1 Jun. 1975 - 30
v. 1975 (Florida State Univ.) 7 p

N76-70822

Unclas
00/98 05340

Introduction

This report covers the six month period from June 1, 1975 to November 30, 1975. For these six months and the following seven months (13 months total) we have received an additional $6000 from NASA to continue our investigation initiated by this grant (NASA NSG-1021). Thus resources at our disposal are rather limited both in terms of support for graduate assistant and computer time. It is anticipated that the Department of Mathematics and the Computer Center of the Florida State University would furnish us some limited computer time when the resources of this grant are depleted.

Large scale disturbances and subsonic jet noise generation

The primary objective of this investigation is to conduct numerical experiments to see if large scale disturbances exist in a turbulent subsonic jet of moderate Mach number and possible noise radiation associated with these flow structures. In the following the formulation of the numerical experiment will be described in some detail. As will be seen below one of the added advantages of numerical experiments is that the intensity of upstream disturbances at the nozzle exit can easily be modified. This allows us the capability of studying numerically the effect of upstream disturbances on jet noise generation with almost no basic change in the computation scheme.

One of the most severe constraints on present day large scale computation is computer core storage requirements. Because of this, it is advantageous to reduce the number of space dimensions that one had to deal with in a particular problem. For the problem under consideration we will assume that there is axisymmetry around the axis of the jet. That is to say, the problem is regarded as independent of the azimuthal angle so that effectively we have a two dimensional problem. Experimentally large scale axisymmetric disturbances often in the form of toroidal vortices have been observed in low Mach number jets. Hence the axisymmetry assumption, although is somewhat restrictive and less general than we would like, does have some experimental support and is not merely a device for reducing one space dimension alone.

The governing equations to be dealt with in the numerical computation are the continuity, momentum and energy equations. If (r,θ,z) are the coordinates of a cylindrical coordinate system with the z axis coinciding the axis of the jet, these equations can be written as

$$\frac{\partial}{\partial t}(r\rho) + \frac{\partial}{\partial r}(r\rho v) + \frac{\partial}{\partial z}(r\rho u) = 0$$

$$\frac{\partial}{\partial t}(r\rho v) + \frac{\partial}{\partial r}(r\rho v^2) + \frac{\partial}{\partial z}(r\rho uv) = -r\frac{\partial p}{\partial r} + \frac{\partial}{\partial r}(r\tau_{rz})$$

$$\frac{\partial}{\partial t}(r\rho u) + \frac{\partial}{\partial r}(r\rho uv) + \frac{\partial}{\partial z}(r\rho u^2) = -\frac{\partial}{\partial z}(rp) + \frac{\partial}{\partial z}(r\tau_{rz})$$

$$\frac{\partial E}{\partial t} + \frac{\partial}{\partial r}(E + rp)v + \frac{\partial}{\partial z}(E + rp) = 0$$

where

$$E = r\rho\left(e + \frac{u^2 + v^2}{2}\right) \quad , \; e = \frac{p}{\rho(\gamma-1)} \quad \text{(internal energy)}$$

$$\tau_{rz} = \tau_{rz}^{(v)} + \tau_{rz}^{(t)} = \text{viscous stress + turbulent shear stress}$$

In the energy equation, the heat transfer rate and energy dissipation due to viscous effect have been neglected. The turbulent shear stress τ_{rz} in the momentum equations will be modelled according to the eddy viscousity concept. An empirical eddy viscousity coefficient will be employed. We would like to emphasis here that we are not solving the turbulence problem. Instead we are merely investigating the nonlinear response of the jet and possible development of large scale disturbances of moderate flow Mach number.

For numerical computation purpose the above equations are somewhat difficult to be used as they are. To eliminate the appearance of the variable r in almost every term of the equations it is advantageous to redefine the dependent variables in terms of mass and momentum fluxes as follows:

$$\frac{\partial \underset{\sim}{U}}{\partial t} + \frac{\partial \underset{\sim}{E}}{\partial r} + \frac{\partial \underset{\sim}{G}}{\partial z} = \underset{\sim}{H}$$

where

[illegible]

$$F = \begin{matrix} m \\ \frac{3-\gamma}{2}\,\frac{m^2}{f} + (\gamma-1)\left(E - \frac{n^2}{2f}\right) \\ \frac{mn}{f} + \gamma\,\tau_{rz} \\ \frac{\gamma m E}{f} + \frac{(1+\gamma)}{2}\,\frac{m(m^2+n^2)}{f^2} \end{matrix}$$

$$G = \begin{matrix} n \\ \frac{mn}{f} + r\,\tau_{rz} \\ \frac{3-\gamma}{2}\,\frac{n^2}{f} + (\gamma-1)\left(E - \frac{m^2}{2f}\right) \\ \frac{\gamma n E}{f} + \frac{1-\gamma}{2}\,\frac{n(m^2+n^2)}{f^2} \end{matrix}$$

We propose to solve this equation by the Lax-Wendroff finite differences scheme. The formulation of the actual finite difference equations including the coordinate transformation, boundary and radiation conditions to be described below is being developed at this time. It is anticipated that this crucial step will soon be completed.

Unlike most analytical methods which have no difficulty in extending the solution to infinity, numerical computation inevitably can only be carried out in a finite bounded region. The choice of this region (the shape of the region) and the boundary conditions prescribed on its perimeter are important factors which will determine the success or failure of the numerical scheme. On the axis of the jet (see figure 1) the appropriate boundary condition is to require the radial velocity component to be zero there. This condition can be deduced from the governing equations or alternatively from axisymmetry consideration. At this jet nozzle exit the Z velocity component is prescribed. This velocity consists of two parts, namely, the steady jet exit velocity and a time dependent component due to upstream turbulent fluctuations. A random number generator subroutine from the computer library will be used to simulate these unsteady fluctuations. Velocity fluctuations with a magnitude of half a percent to 4% of the mean jet exit velocity are believed to be typical of most jets used in laboratory experiments, and will be used in the computation. On the remaining portion of the boundary a radiation condition will be prescribed. In other words, only outgoing disturbances are allowed there.

In a jet flow, large velocity gradient exists in the mixing layers. These regions are highly unstable. This shear layer instability is believed to be responsible for the generation (at least during the initial stages of development) of large scale disturbances. Therefore adequate resolution must be provided in a computation scheme for this important region of the jet flow. To do this we intend to adopt the following coordinate transformation from the (r,z) plane to the (ζ,η) plane,

$$\zeta = \alpha_1 + \beta_1 \tan^{-1}(z - \tan \frac{\alpha_1}{\beta_1})$$

$$\eta = \alpha_2 + \beta_2 \tan^{-1}(\frac{r}{b(z)} - \tan \frac{\alpha_2}{\beta_2})$$

where $\alpha_1, \alpha_2, \beta_1$ and β_2 are constants. $b(z)$ is the half width of the jet measuring from the jet axis to the half velocity point. This transformation magnifies the shear layer region thus giving the proper emphasis there. The inverse transformation is

$$z = \tan(\frac{\zeta - \alpha_1}{\beta_1}) + \tan \frac{\alpha_1}{\beta_1}$$

$$r = b(z)[\tan \frac{\eta - \alpha 2}{\beta_2} + \tan \frac{\alpha_2}{\beta_2}]$$

The computation will be carried out in the (ζ,η) plane. A rectangular grid of 250 x 50 points will be used. The corresponding physical plane is not rectangular but can easily be found from the inverse transformation.

In going from the jet flow to the acoustic field there is a substantial change in the magnitude of the flow variables. This non-uniformity of the sizes of the computed variables could be a potential source of problems. It is possible that the round-off errors in computing the jet flow are not small compared to the acoustic field making the meaning of the calculated noise intensity somewhat ambiguous. To avoid this problem we intend to separate a large part of the mean flow from the actual computation. An appropriate mean flow profile will be determined using an integral procedure. In our computation only the difference between the actual flow and this approximate mean flow is being calculated. In this way the sizes of the computed variables would not differ too greatly in the whole solution plane.

The above consideration essentially completes the formulation stage of our numerical experiment. During the next seven months our efforts will be concentrated in setting up the finite difference equations and boundary conditions. They will then be programmed and tested in the FSU CDC 6500 computer. This step is believed to be one of the most time consuming part of the project in terms of man hours and to a certain degree computer time as well.

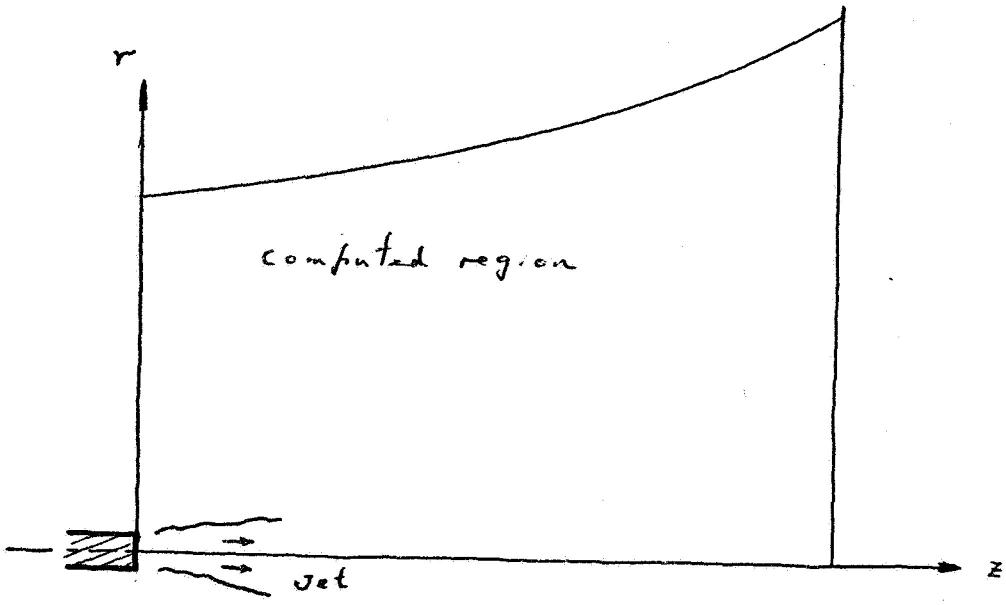

Figure (1) Physical Plane